

# FreeCAD-CFD Workbench

## Tutorial 3: Boundary layer meshing and turbulent flow



# CFD Workbench

## WORKBENCH

This workbench aims to help users set up and run CFD analysis. It guides the user in selecting the relevant physics, specifying the material properties, generating a mesh, assigning boundary conditions and setting the solver settings before running the simulation. Where possible best practices are included to improve the stability of the solvers.

## INSTALLATION

### WINDOWS:

- <https://www.freecadweb.org/wiki/Download>
- Install CfdOF from Tools | Addon manager
- Go to Edit | Preferences | CFD to check and install dependencies

### LINUX:

- [https://www.freecadweb.org/wiki/Install\\_on\\_Unix](https://www.freecadweb.org/wiki/Install_on_Unix)
- Install CfdOF from Tools | Addon manager
- Install OpenFOAM (5.0 recommended) (<https://openfoam.org/download/>)
- Install Paraview (tested with 5.0.1)
- Optional - Install GMSH (optional, 2.13+)
- Go to Edit | Preferences | CFD to check dependencies and install cfMesh

## LATEST INFORMATION

Please see the CfdOF [README file](#) for up-to-date information.

## LEAD DEVELOPERS

Johan Heyns (CSIR, 2016-2018) [jaheyns@gmail.com](mailto:jaheyns@gmail.com),

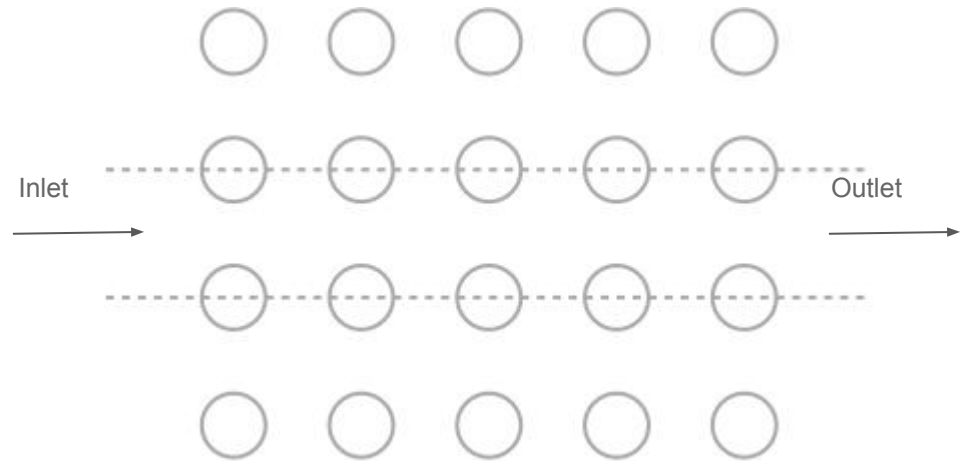
Oliver Oxtoby (CSIR, 2016-2018) [oliveroxtooby@gmail.com](mailto:oliveroxtooby@gmail.com),

Alfred Bogaers (CSIR, 2016-2018) [abogaers@csir.co.za](mailto:abogaers@csir.co.za),

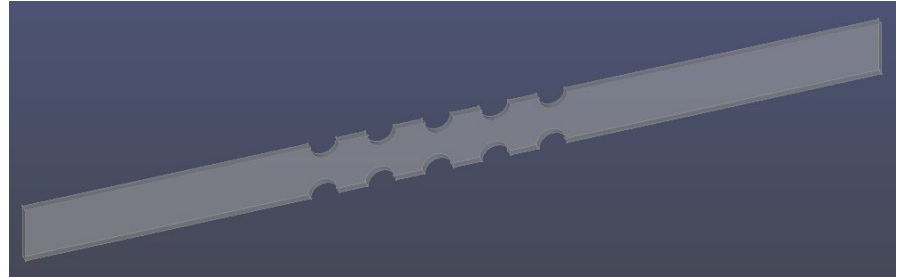
# Part Design

# Tube bundle

- To demonstrate how to model viscous flow, the pressure drop over a tube bundle is calculated.
- Bundle parameters
  - Tube diameter: 100 mm
  - Tube pitch: 200 mm
- Assuming symmetric conditions we model the flow through one of the channels.



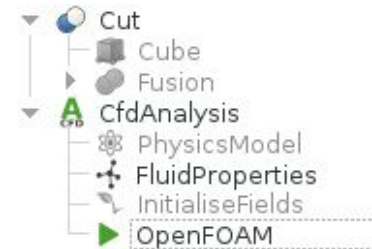
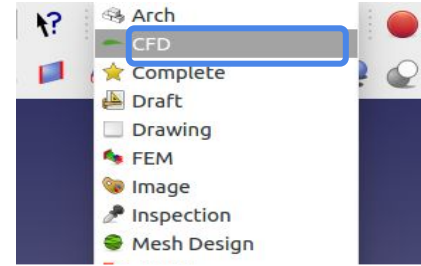
- Activate the “Part” workbench
- Create a cube
  - Length: 3000 mm
  - Width: 200 mm
  - Height: 25 mm
- Create 10 cylinders
  - Diameter: 100 mm
  - Spacing: 200 mm
- For the channel cut the cylinders away from the cube



# Boundary layer meshing

# Initialise CFD analysis

- Activate the CFD WB by clicking on the dropdown menu in the taskbar and select “CFD”
- Create a CFD analysis which will automatically generate:
  - Physics model
  - Fluid properties
  - Initialise fields
  - OpenFOAM solver

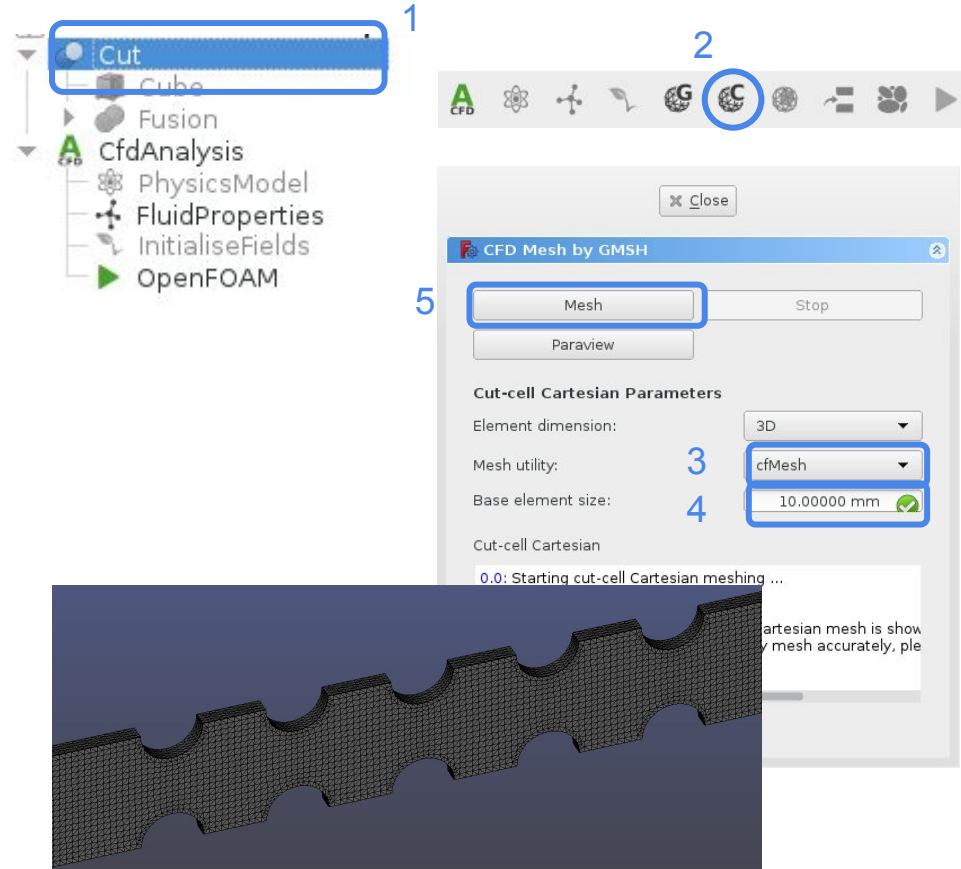


# Create an initial mesh

- To create a preliminary mesh select the 'Cut' object.
- Click on the cut-cell Cartesian icon
- Select the 'cfMesh' utility which allows for boundary layer meshing.
- Set the maximum element characteristic length to 10 mm and mesh.
- You should now have a uniformly spaced cut-cell Cartesian mesh.


NOTE: Currently, only the 'cfMesh' utility supports boundary layer meshing.

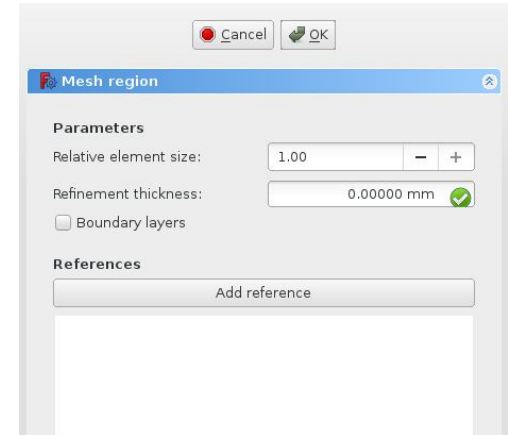
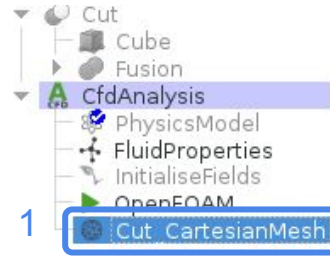
NOTE: See Tutorial 1 for information on how to create a 2D mesh





# Adding boundary layers

- Highlight the mesh object to activate the mesh region icon 
- Create a mesh region by clicking on the icon .
- A mesh region object will now have been created (if not visible expand “Cut\_CartesianMesh”).
- Double click it to open the task panel.
- In this task panel, we can now edit the refinement and boundary layer parameters.

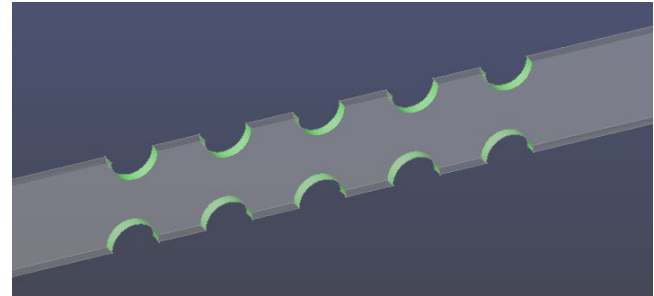
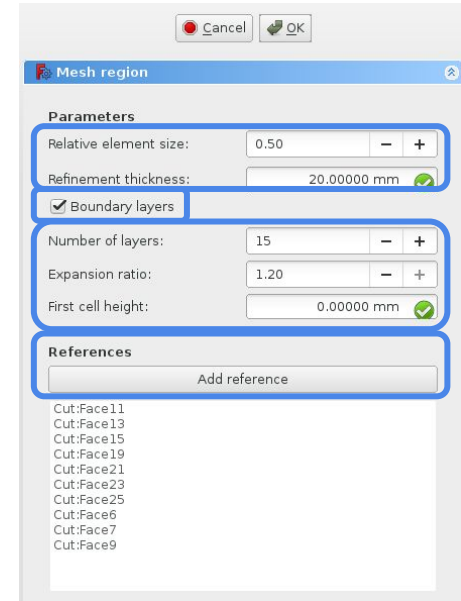


# Mesh region parameters

- The refinement parameters ‘Relative element size’ and ‘Thickness’ are respectively used to set the cell size relative to the base mesh and the absolute thickness of the refinement region.
  - Relative element size: 0.5
  - Refinement thickness: 20 mm
- Check the “Boundary Layer” box
- The user can set the number of layers, the expansion ratio that governs the growth of the layers and, optionally, the maximum first cell height (The default 0 mm will be ignored).
  - Number of layers: 15
  - Expansion ratio: 1.2
- Next, the reference faces need to be added.

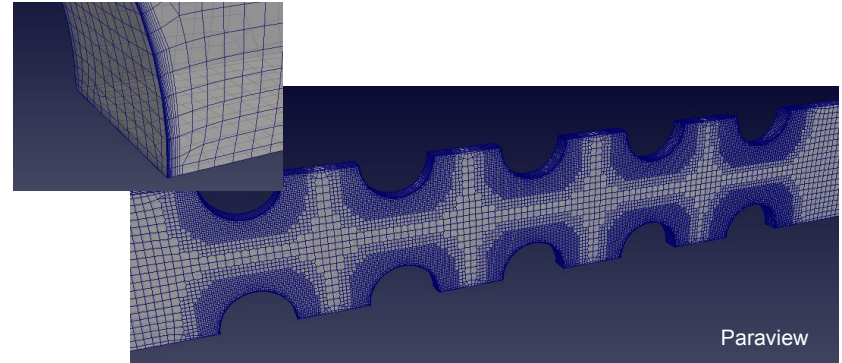
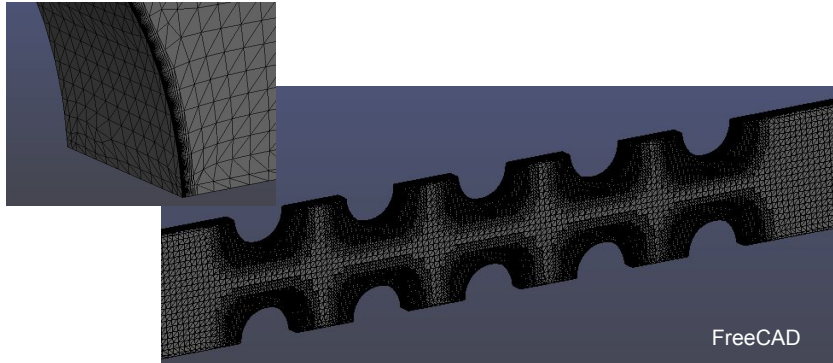
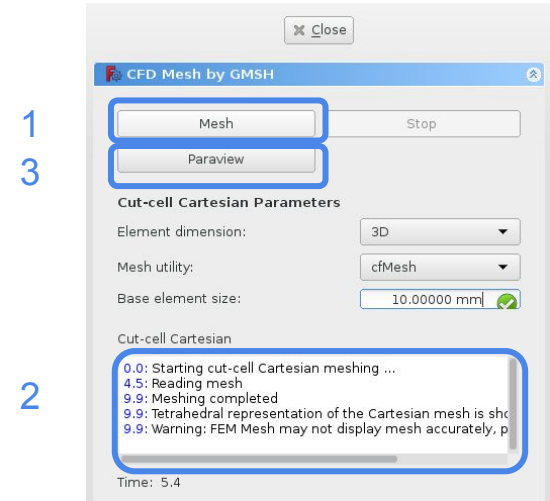
NOTE: The expansion ratio is limited to be greater than 1.0 and less than 1.2.

1  
2  
3  
4



# Update mesh

- Once the mesh region parameters are updated the user can go back to the mesh task panel and click on 'Mesh' to recompute.
- The message console shows the progress and once meshing is completed a tetrahedral representation of the mesh is displayed.
- For a more accurate representation of the mesh click on "Paraview" to open the viewer.



# Viscous analysis

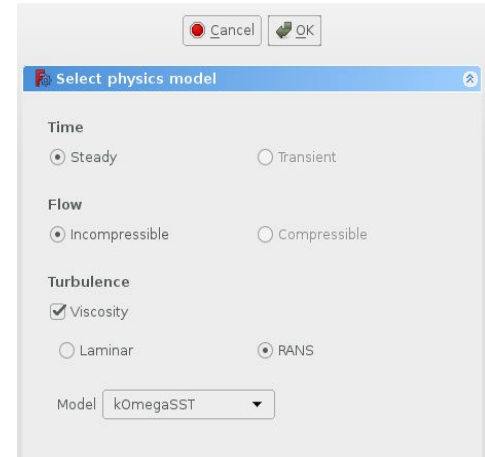
# RANS analysis

- In the “PhysicsModel” task panel under “Turbulence”, select the RANS checkbox which will automatically select the k-w SST model.

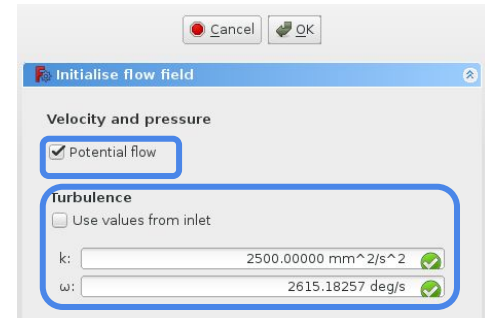
NOTE: Currently the CFD WB only supports the k-w SST model.

- Activate the “InitialiseFields” task panel and ensure the “Potential flow” checkbox is selected. The high aspect ratio boundary layer cells require a well initialised velocity field.
- The k and w can be initialised using the values from the inlet or specified by the user.

1



2



3

# Boundary conditions

- In addition to entering the velocity at the inlet boundary, the user is required to specify the turbulence quantities.
- The user can either specify the Turbulence intensity and Length scale or the Kinetic energy (k) and Dissipation rate (w).

NOTE: The following links might be useful in guiding the user in specifying the inlet turbulent quantities:

<https://turbmodels.larc.nasa.gov/sst.html>

[https://www.cfd-online.com/Wiki/Turbulence\\_intensity](https://www.cfd-online.com/Wiki/Turbulence_intensity)

[https://www.cfd-online.com/Wiki/Turbulence\\_length\\_scale](https://www.cfd-online.com/Wiki/Turbulence_length_scale)

- For the test problem, we will use turbulent intensity
  - Turbulent intensity: 5 %
  - Length scale: 2 mm



**Turbulence specification**

Intensity & Length Scale ▼

turbulence intensity and eddy length scale

Turbulence intensity (I) 0.05000 ✓

Length scale (l) 2.00000 mm ✓

**Turbulence specification**

Kinetic Energy & Specific Dissipation Rate ▼

k and omega specified

Turbulent kinetic energy (k) 1080.00000 mm<sup>2</sup>/s<sup>2</sup> ✓

Specific dissipation rate (ω) 57.29578 deg/s ✓

# Boundary conditions

- Viscous/no-slip boundaries should be applied to the tube walls.
- For no-slip walls, wall functions are used for the turbulent quantities. This requires the  $y^+$  of the first cell to be between 30 and 150.

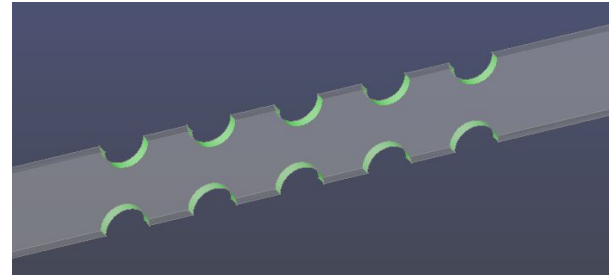
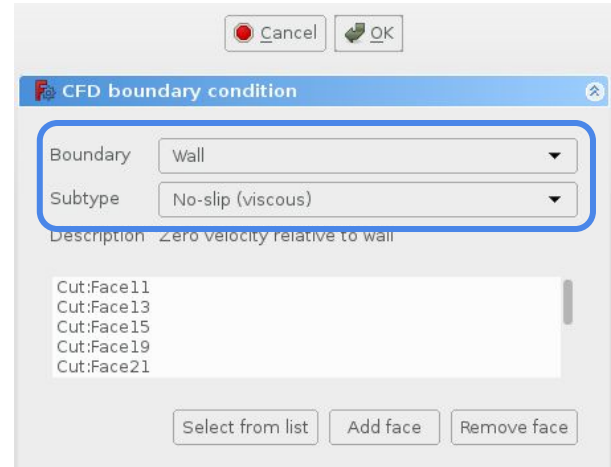
NOTE: The following links might be useful in guiding the user to compute the non-dimensional  $y^+$  value:

[https://www.cfd-online.com/Wiki/Dimensionless\\_wall\\_distance\\_\(y\\_plus\)](https://www.cfd-online.com/Wiki/Dimensionless_wall_distance_(y_plus))

[https://www.cfd-online.com/Wiki/Law\\_of\\_the\\_wall](https://www.cfd-online.com/Wiki/Law_of_the_wall)

- The remainder of the walls may be treated as either “Slip” or “Symmetric” and “Static pressure” is prescribed at the outlet.

NOTE: An undefined boundary patch will default to slip/inviscid wall.



# Running the simulation

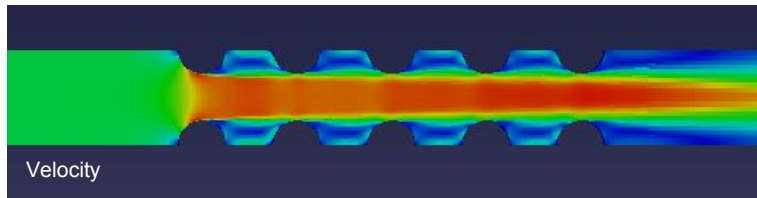
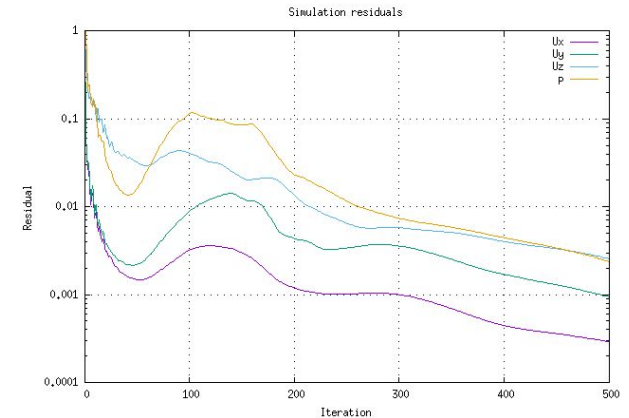


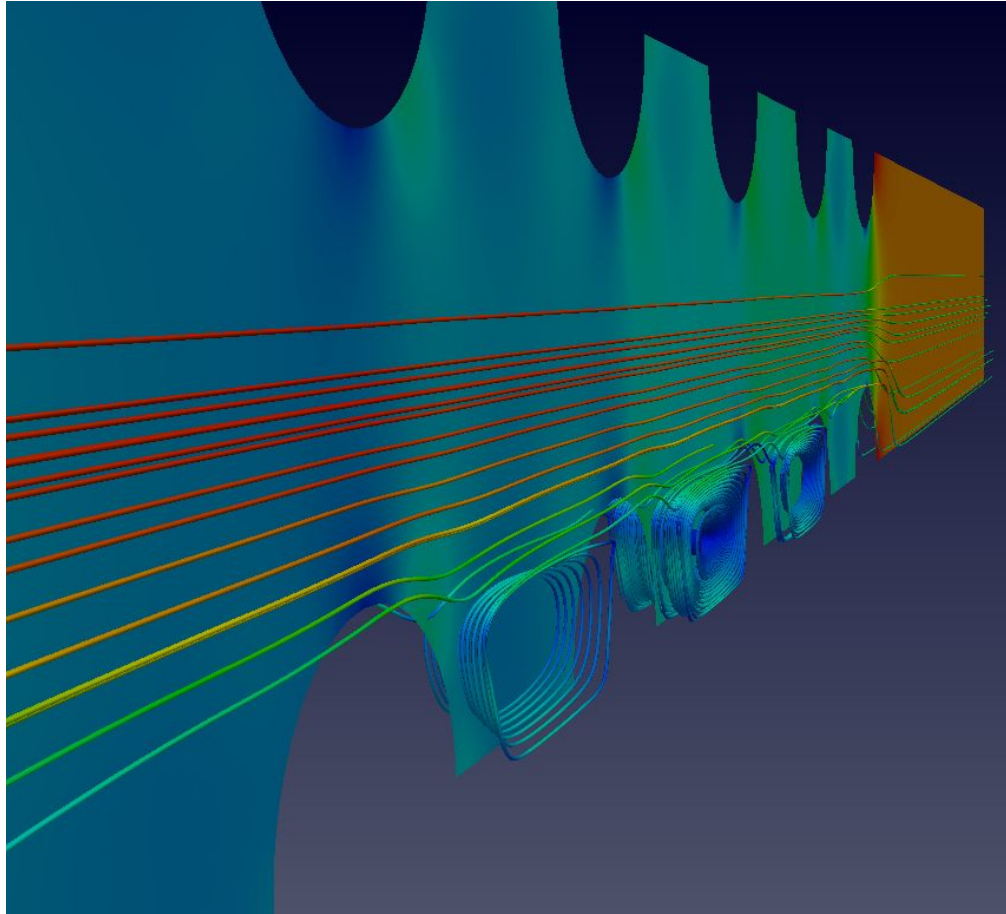
# Simulation

- In the “OpenFOAM” data panel, the parallel setting can be set to true to run over multiple cores.
- The solver Task panel may then be activated, from which the case is written and simulation run.
- The residual plot is automatically created and the results can be viewed in Paraview.

NOTE: The incompressible solver in OpenFOAM stores the kinematic pressure,  $p = P/\text{density}$ .

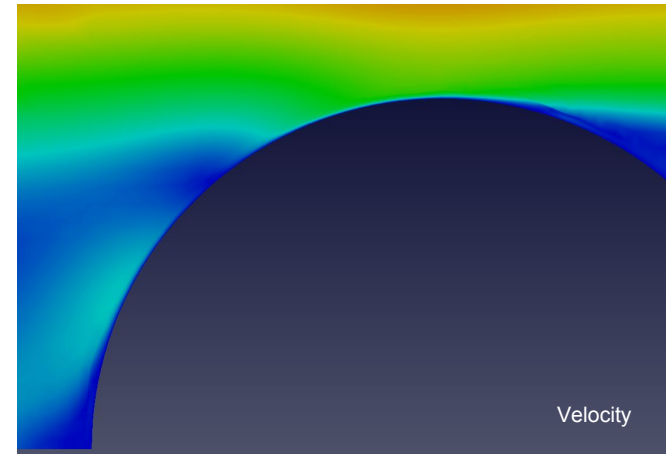
Solver	
Input Cas...	case
Parallel	true
Parallel C...	4
Solver N...	OpenFOAM
Working Dir	/tmp
Time Ste...	
Converg...	0.00010
End Time	500.00000
Time Step	1.00000
Write Inte...	10.00000





Paraview:

- “Slice” to show the pressure contours.
- “StreamTracer” to show the streamlines
- “Tube” to apply thickness to the streamlines



The End